Multi-objective optimization of rotary device bracket for automatic diesel cylinder block line based on ANSYS analysis

ABSTRACT

Slewing device is the key component of diesel engine cylinder block automatic line, whether its design is reasonable affects the performance of the whole automatic line.In order to enhance its structural rationalization and economic efficiency,taking a certain type of cylinder block automatic line slewing device as the research object, firstly, use Solid Works to carry out three-dimensional modeling, and then use ANSYS Workbench platform to establish finite element models,determine the material properties, constraints and loads of its model, perform static and modal analysis of its bracket to understand its dynamic and static performance, and optimize its dimensions by combining with multi-objective optimization theory,the results show that the optimization effectively reduces the mass of the slewing unit bracket and improves the dynamic and static performance.

Keywords:slewing device; bracket；finiteelement；multi-objective optimization

1. INTRODUCTION

Diesel engine is the heart of the car, and the cylinder block is an important part in the diesel engine, its manufacturing technology is also significantly improved1[1]. China's agricultural diesel locomotives from a single cylinder to multi-cylinder, functional coverage is very wide, the product variety is extremely large, has become a veritable world power in the manufacture of agricultural diesel engines[2].In the modern society of domestic machinery technology development and processing research and development of material and technological conditions of the level of rapid development of the background, the market to reduce production costs, shorten the manufacturing cycle and improve product performance and other needs are becoming increasingly strong, cylinder automated production line has become the trend of the diesel engine manufacturing industry[3].Diesel engine manufacturing process, its cylinder block generally use processing automatic line to automatic processing is completed[4]. In the diesel engine cylinder block processing automatic line, the time and work of the rotary movement of the proportion of the work cycle in the entire processing automatic line occupies a pivotal position, so the rotary device is a cylinder block automatic production line is extremely important part of it in different working conditions under different sizes of the load[5], the efficiency of the entire cylinder block automatic line by the structure of the safety and stability of the impact, so the diesel engine Therefore, the multi-objective optimization of the rotary device of diesel engine block automatic production line is indispensable.

In recent years, the development of the computer is rapidly changing, a variety of finite element analysis software and optimization algorithms in the field of machinery and equipment is also gradually emerging, more and more companies are beginning to increase innovation, finite element simulation technology is applied to the structural optimization of the equipment research and development and production of the various stages of the method is not only largely improve the efficiency of the product design, but also reduces the number of prototypes and modifications to reduce the cost of production and promote enterprise product innovation and development capabilities[6].This method not only greatly improves the efficiency of product design, but also reduces the number of prototypes and modifications, thus reducing the production cost and promoting the product innovation and development ability of enterprises. There are numerous finite element simulation systems on the market, of which ANSYS Workbench is currently the most used CAE simulation and analysis software. In this paper, the author establishes a geometric model of the stent of the cylinder automatic line tipping device, and carries out multi-objective optimization design of the stent according to the results of finite element analysis, which ultimately makes the parts fully meet the requirements of institutional strength and stiffness, and realizes structural lightweighting, improves the first-order intrinsic frequency, and greatly reduces the production cost, and provides a solution for the design and optimization of the cylinder automatic line tipping device under other actual working conditions. Idea[7].

2. STRUCTURAL DESIGN OF THE ROTARY UNIT

The slewing device is used to move the table and the slewing disk stably in rotation, and it is mainly composed of self-propelled roller conveyor, slewing disk, motor gear, bracket and other parts. As shown in Fig.1, the upper part of the rotary device of the diesel engine cylinder block machining automatic line is the conveying part, and the cylinder block element is located above the conveying roller conveyor. The chain-driven conveying motor is located in the left half of the conveying part, and the motor is mounted in the motor mounting bracket for fixation and protection[8].The motor shaft rotates, driving the sprockets to rotate and engage the chain, thereby rotating and rolling the chain. The chain rolls by engaging the sprocket on the shaft, which turns the drive shaft. The drive shaft is fixed on the roller conveyor bracket through the profile, and the process of shaft rotation drives the cylinder material above it to be conveyed forward through static friction. Diesel engine cylinder in the forward conveying process, the cylinder itself has a large self-weight, and thus has a large inertia; at the same time, the conveying distance is farther, requiring a larger conveying speed[9].In order to ensure that the diesel engine cylinder can be accurately stopped in a certain position, can be accurately rotated 90 degrees, set up a speed measurement at the end of the end of the photoelectric switch, control the start and stop of the conveyor belt and speed10.

The rotary part of the diesel engine cylinder block machining automatic line rotary device is located in the lower part of the whole device, and its composition is relatively simple, and its main parts are rotary motor, turntable bearing, pinion and bracket. Pinion and turntable bearing are the core parts to realize the rotary function. Starting the motor, the motor shaft rotates, and there is a flat keyway on the motor shaft, which transmits the power to the pinion gear through the flat key connection[10].This design adopts A type ordinary flat key as the type of sprocket and shaft connection. The motor shaft rotates at a high speed, and the pinion gear may be displaced, which may affect the transmission and lead to unstable transmission process and other dangerous situations. In order to ensure that the power transmission process is stable and the pinion position does not change, the pinion end face is fastened to the end cap using a screw connection. The pinion gear engages with the rotary bearing with internal teeth, the rotary bearing is closely connected with the upper part of the conveying part, and the pinion gear transmits the power to the slewing bearing in an engaging manner to drive it to rotate, thus driving the entire conveying part to rotate, i.e., driving the cylinder element to rotate[11].After the cylinder element completes the clockwise rotation of 90 degrees, the lifting and tilting device comes to this device to grab the workpiece cylinder element and enter the next section.

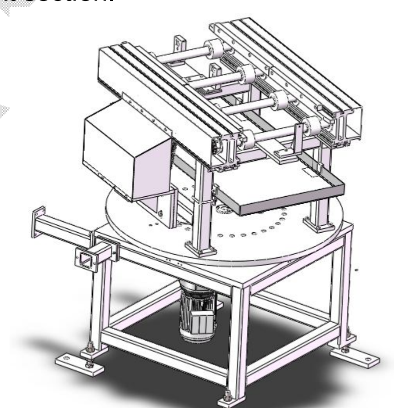


Fig.1.Slewing unit model

3.FINITE ELEMENT MODELING OF THE TIPPING DEVICE

As the upper part of the rotary device is installed with self-propelled rollers, rotary discs, workpieces and other parts, the bracket carries a large load, in order to extend the service life of the rotary device, it is necessary to optimize the design of the bracket of the rotary device, due to the existence of chamfers, threaded holes and other small features of the three-dimensional model, which have little impact on the performance of the entire rotary device, therefore, the bracket of the rotary device in the design of the structure of the model to do a partially simplified[12].The simplified three-dimensional model is shown in Fig.2.

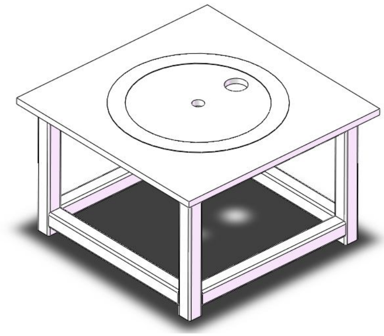


Fig.2.Stent model

3.1 Adding Material Properties

Using the information interaction between the 3D modeling software SolidWorks and the finite element analysis software ANSYS Workbench, the 3D model is opened through the finite cloud software, the geometric structure is generated in DesignModeler, and the bracket defines the material as Q235 steel, and the corresponding material performance parameters are shown in Table 1.

Table 1. Material properties

Parameters	Numerical Value
Density /(g.cm^3)	7.85
Elastic modulus /GPa	210
Poisson's ratio	0.27
Yield Strength/ Mpa	235

3.2 Defining Contact Between Parts

Bracket is composed of multiple parts of the assembly, they work in the rotary device does not occur between the relative motion, it can be understood that the parts of each surface is connected together by welding, in the finite element analysis does not take into account the influence of the weld, and then according to the assembly of the relationship between the parts of the parts of the contact between the surface of the definition of the bonded (bonded) way[13].

3.3 Stent Meshing

There is a very close relationship between the mesh quality and the simulation results, in order to effectively avoid the stress concentration phenomenon in the simulation process and to improve the computational speed, it is easy to get a better mesh quality by using the free mesh division methodthe mesh cell size is set to 10mm, the mesh division is performed on the model, and a total of 245136 nodes are generated with 104134 cells, the mesh map is divided as shown in Fig. 3.[14]

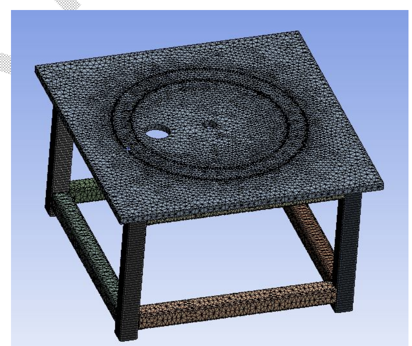


Fig.3.Meshing of the model

3.4 Load Application and Setting of Boundary Conditions

A certain type of diesel engine cylinder block automatic line working environment is not bad, taking into account the cylinder block can be processed at multiple angles, the upper end of the bracket is installed with a rotary disk, capable of 180 degrees of rotation of the cylinder block, so in the process of cylinder rotation, the bracket will be subjected to the rotary disk, the workpiece gravity, etc., the role of the bracket and the rotary disk above the contact surface of the bracket to add the size of the bracket for the gravity of the g = 9.8066 m / s2 , and the combined force is calculated to be about 33,358.04 N, which is equally divided at the top of the bracket where the rotary disk is mounted to simulate the force when the rotary device is actually working. The slewing bracket is connected to the ground through four square tubes, so it is necessary to fix the four square tubes and add a fixed support at the bottom of the square tubes, as shown in Figure 4.

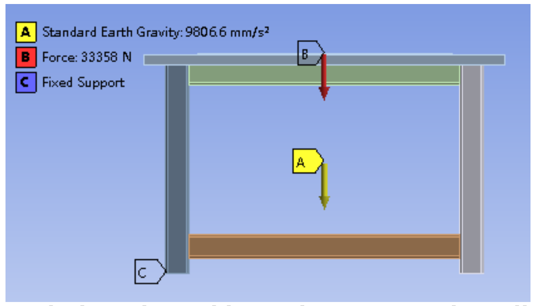
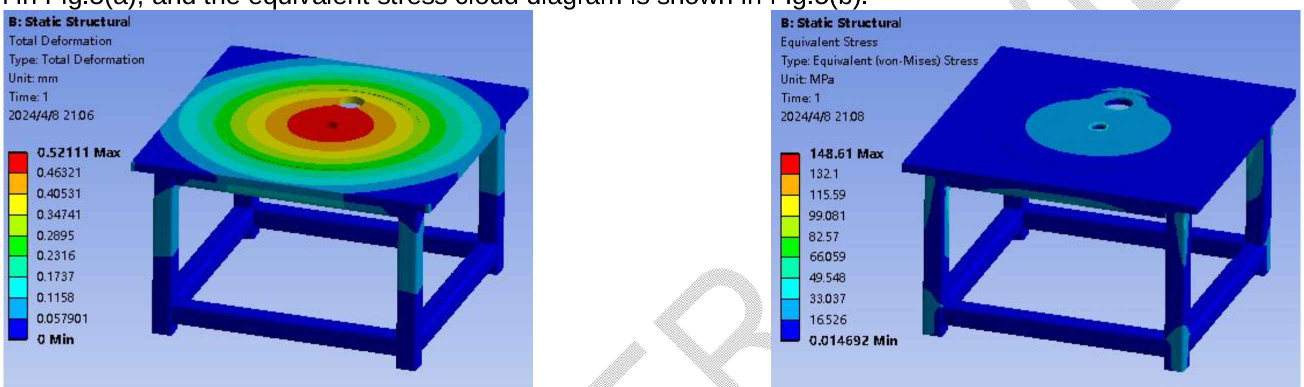


Fig.4.Applied loads and boundary constraints diagram

3.5 Finite Element Analysis Results

3.5.1 Static analysis

After applying the loads and boundary conditions, ignoring the inertia and damping between the parts, the effect of the loads on the device becomes a hydrostatic analysis[15]. After the load application and boundary condition setting, finite element solution can be carried out for the slewing device bracket, and its total deformation cloud diagram is obtained as shown in Fig.5(a), and the equivalent stress cloud diagram is shown in Fig.5(b).



（a）total deformation cloud diagram （b）equivalentstress cloud diagram

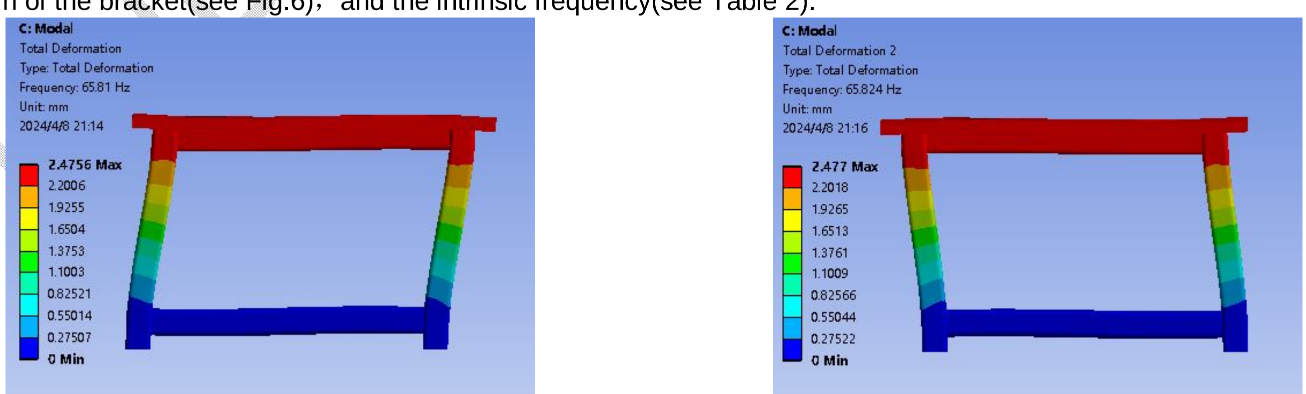
Fig.5.Total deformation cloud diagramandequivalent stress cloud diagram

According to the figure, it can be seen that the maximum total deformation occurs in the contact area between the rotary disk and the bracket, and the maximum deformation is 0.45512 mm, while the maximum equivalent stress occurs in the square tube connection below the bracket top plate, and the size is 139.2 MPa. Because the stress concentration mainly exists at the geometry mutation, and the maximum deformation is less than 1 mm, the model can satisfy the actual design requirements.

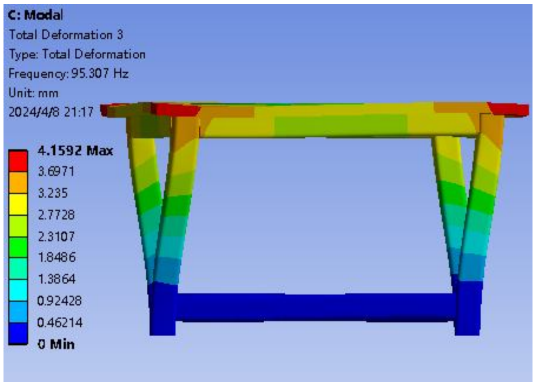
3.5.2Modal analysis

Modal analyses are usually performed on linear systems in which the structural dynamics are characterized by equations or coordinates and are widely used in industrial manufacturing.

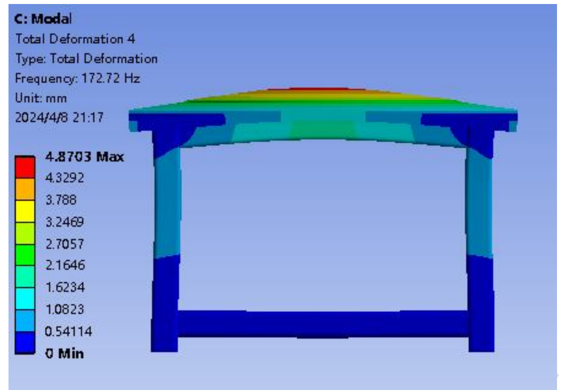
Due to the high requirements for the machining accuracy of the cylinder block, any small vibration that occurs during the work of the cylinder block automatic line may cause the position of the cylinder block to change, resulting in machining errors, and the advantages and disadvantages of the rotary device can be evaluated through modal analysis to provide a theoretical basis for the vibration analysis and optimization of the design, and to avoid the occurrence of resonance in the work[16]. Establish the model of the bracket on SolidWorks platform, and then import it into ANSYS Workbench software, after adding the fixed support, the modal analysis of the model can be derived from the 1st-6th order modal vibration pattern of the bracket(see Fig.6)，and the intrinsic frequency(see Table 2).



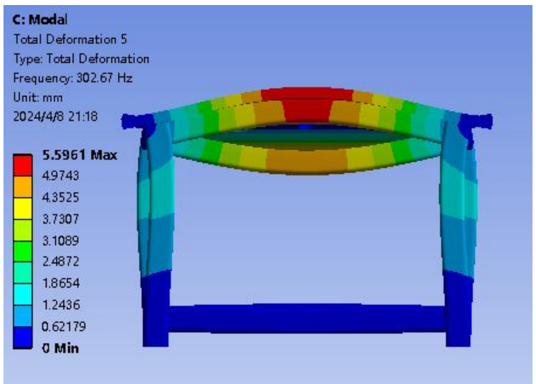
（a）1st order vibration mode *（b）2nd order vibration mode*



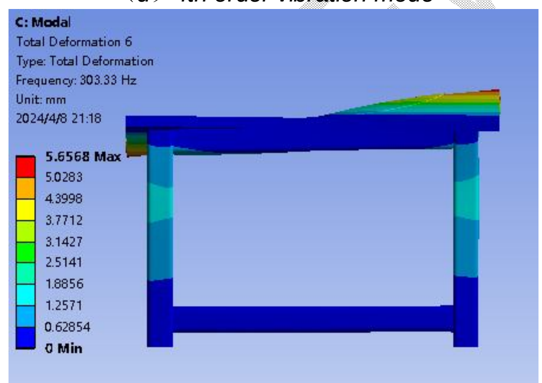
（c）3rd order vibration mode



（d）4th order vibration mode



（e）5th order vibration mode



（f）6th order vibration mode

Fig.6.1st-6th order modal vibration pattern of the bracket

Table 2.1-6th order intrinsic frequency

Modal Order	Solid Frequency /Hz
1	65.81
2	65.824
3	95.307
4	172.72
5	302.67
6	303.33

Animation allows dynamic demonstration and animation output of the results, which can more intuitively observe the change process of the bracket at different frequencies. Since the engine speed of the cylinder block automatic line slewing device is 28 r/min and the frequency is 1.67 Hz, it can avoid the resonance frequency of the bracket and has good safety performance.

4 MULTI-OBJECTIVE OPTIMIZATION

4.1Response Surface Modeling

Response surface method is a method developed on the basis of experimental design method, and its core idea is to simulate the variables and objective functions in the experimental design by constructing a model to predict the response values of non-experimental points[17].Design Explorer, as a fast optimization tool in AWE environment, combined with statistical methods, is able to form a response surface based on a limited number of experimental design point parameters. In this process, the number of design points can be increased to improve the model accuracy.

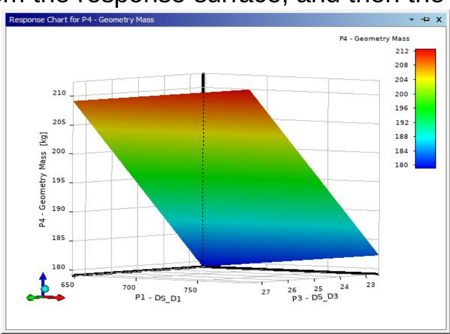
Design Explorer, as a fast optimization tool in the AWE environment, combined with statistical methods, is able to form response surfaces based on a limited number of experimental design point parameters. In this process, the number of design points can be increased to improve the model accuracy[18].

In practice,the stent can be parametrically modeled using SolidWorks software,and the three parameter dimension names DS_D1(base square tube length),DS_D2(diameter of the contact surface between the rotating disk and the stent),and DS_D3(thickness of the stent's top plate) can be set as the input parameters P1,P2,P3 of the ANSYS Workbench through the global variable,so as to realize the association of each parameter.Open the Response Surface Optimization method in the finite element analysis software,enter the upper and lower limits of DS_D1,DS_D2,and DS_D3 as 10% of the default values in ANSYS Workbench,andselect the intermediate composite design method to generate 15 groups of experimental design points,as shown in Table3.The corresponding output parameters are P4(geometry mass),P5(maximum total deformation),P6(maximum equivalent stress) and P7(1st order vibration frequency).

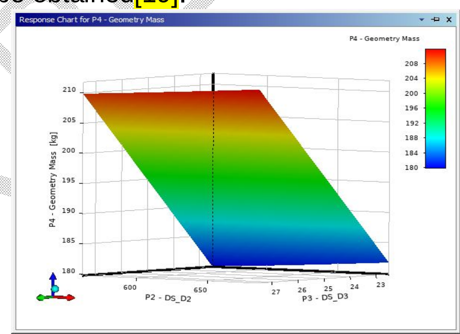
Table 3.Experimental design

Number	P1/mm	P2/mm	P3/mm	P4/kg	P5/mm	P6/MPa	P7/Hz
1	720	624	25	195.75	0.521	148.61	65.81
2	648	624	25	194.11	0.446	138.33	66.139
3	792	624	25	197.38	0.607	161.15	65.035
4	720	561.6	25	194.83	0.568	156.41	65.964
5	720	686.4	25	196.75	0.477	142.37	65.638
6	720	624	22.5	180.68	0.623	163.68	68.182
7	720	624	27.5	210.81	0.443	136.97	63.695
8	661.46	573.27	22.967	181.43	0.572	157.32	68.137
9	778.54	573.27	22.967	184.08	0.732	177.29	67.199
10	661.46	674.73	22.967	182.99	0.488	144.41	67.87
11	778.54	674.73	22.967	185.64	0.635	164.73	66.902
12	661.46	573.27	27.033	205.92	0.433	135.9	64.458
13	778.54	573.27	27.033	208.57	0.549	152.86	63.615
14	661.46	674.73	27.033	207.48	0.375	125.42	64.243
15	778.54	674.73	27.033	210.13	0.483	142.76	63.369

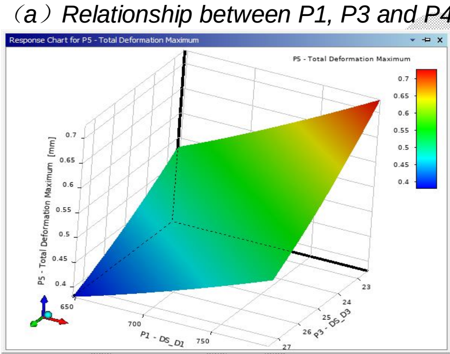
Before the response surface analysis, in the meta-model, the genetic aggregation method is selected to simulate the construction of the response surface model, which makes the fitting accuracy more accurate, and after the calculation, the response chart mode is changed to 3D, as shown in Fig.7, which are the geometric structural quality of the model, the maximum deformation of the model, the maximum equivalent stress, and the critical response surface of the 1st order intrinsic frequency, from which the objective function can be observed to determine the optimal conditions or optimal region from the response surface, and then the optimization scheme can be obtained[19].



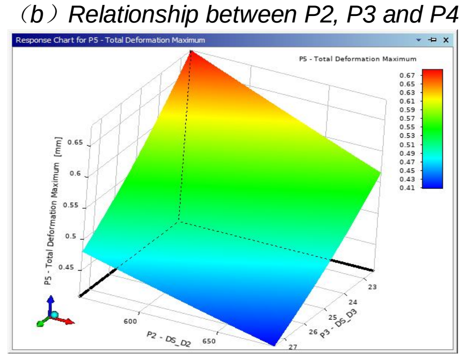
（a）Relationship between P1, P3 and P4



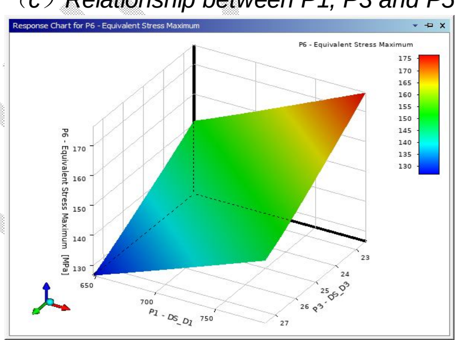
（b）Relationship between P2, P3 and P4



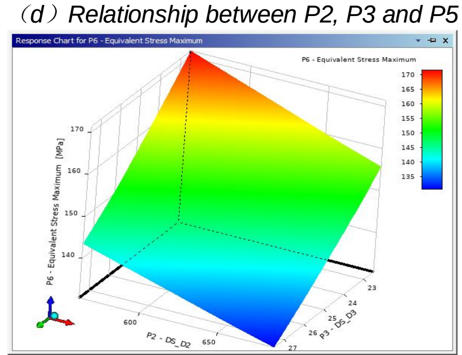
（c）Relationship between P1, P3 and P5



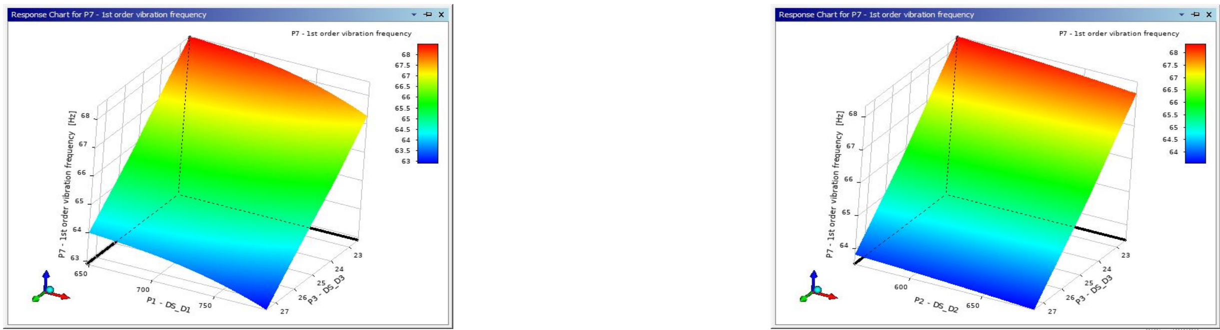
（d）Relationship between P2, P3 and P5



（e）Relationship between P1, P3 and P6



（f）Relationship between P2, P3 and P6



（g）*Relationship between P1, P3 and P7* （h）*Relationship between P2, P3 and P7*

Fig.7.Response surface modeling

4.2Sensitivity Analysis

Sensitivity analysis is an important part of modern design theory to evaluate the effects of changes in design variables or parameters on the response characteristics of a structure. In the Response Surface Optimization module, both local and global sensitivity analysis methods are provided to help achieve further optimized design solutions[20].

The local sensitivity analysis (LSA) enables the sensitivity of individual output values to input values to be calculated, thus eliminating certain input parameters that do not have a significant impact on the optimization results, as shown in Fig.8, where P1 and P4 have a greater impact on the experimental results compared to P2. Global Sensitivity Analysis (GSA) is used to complete the optimization process by selecting the best optimized design point from the candidate points for subsequent comprehensive response surface analysis.

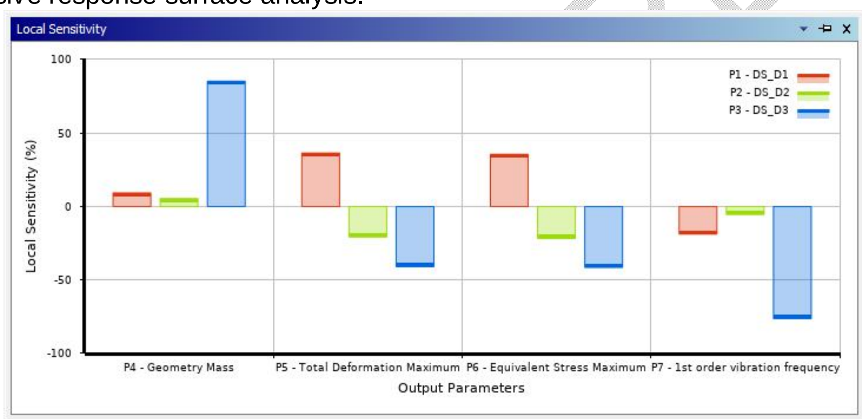


Fig.8.Local sensitivity analysis

4.3Sensitivity Analysis

Optimal design is the application of mathematical theories and optimization techniques to the design of industrial products in order to select the best result among multiple feasible solutions. Since the arrangement of the cylinder automatic line needs to be reasonable and efficient, this paper takes the lightweighting of the bracket, i.e., the minimization of the model mass, as the optimization objective, and also sets the maximization of the first-order intrinsic frequency as the second optimization objective, in which, after the analysis of the software system, three sets of candidate design points are given, as shown in Fig.9. Fig.10 shows the global sensitivity distribution of the optimized design, according to the sensitivity analysis and comprehensive consideration, two groups of candidate points are selected as the design points for this optimization, and the final input and output values obtained after dimensional optimization, as shown in Table 4. The project analysis flowchart of the whole structure optimization is shown in Fig.11.

Candidate Points

	Candidate Point 1	Candidate Point 2	Candidate Point 3
P1 - DS_D1	648.02	648.01	649.47
P2 - DS_D2	561.62	562.22	561.69
P3 - DS_D3	22.5	22.5	22.5
P4 - Geometry Mass (kg)	178.14	178.15	178.18
P5 - Total Deformation Maximum (mm)	0.5859	0.58534	0.58768
P6 - Equivalent Stress Maximum (MPa)	158.67	158.59	158.92
P7 - 1st order vibration frequency (Hz)	68.616	68.615	68.613

Fig.9.Candidate results

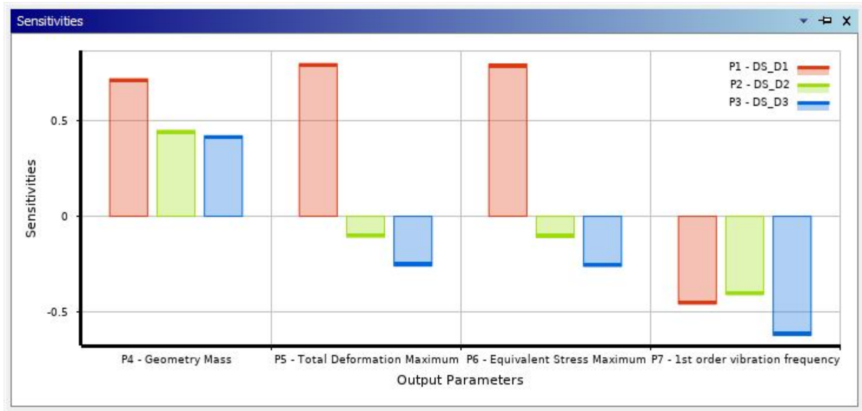


Fig.10.Global sensitivity analysis

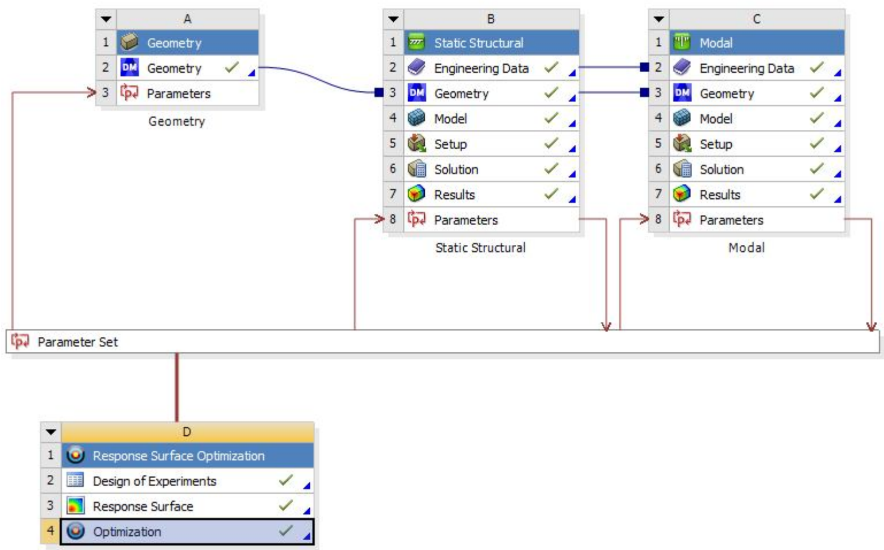


Fig.11.Project Analysis Flowchart

Table 4.Stent Input and Output Variables and Optimization Design Objectives

Parameter Type	Designation	Original Design	Optimization Goals	Optimization Results
importation	P1/mm	720	/	648.02
	P2/mm	624	/	562.13
	P3/mm	25	/	22.501
exports	geometry mass/kg	201.56	/	0.513
	maximum total deformation/mm	0.455	/	142.44
	maximum equivalent stress/MPa	139.2	maximum	68.471
	1st order vibration/Hz	65.547	minimum	183.66

After optimization, the mass of the rotary device bracket is 183.66kg, which is 17.9kg less than 201.56 before optimization, and basically achieves the expected goal; the 1st order vibration frequency of the geometric model is increased from 65.547Hz to 68.471Hz, which makes the system able to meet the working requirements; in addition, the maximum deformation of the rotary device after optimization is 0.513mm, which is basically unchanged compared with 0.455mm before optimization. making full use of the performance of the material and shortening the design cycle[21]; the maximum equivalent force is increased from 139.2MPa to 142.44MPa, and the permissible stress [σ]=195.83MPa is still greater than the maximum equivalent force according to the yield strength of Q235 steel 235MPa, with the safety coefficient of 1.2, and the system safety requirements have been completely met. The safety requirements of the system are met.

5.CONCLUSION

In this paper, ANSYS Workbench software is used to carry out static and dynamic finite element analysis of a certain type of diesel engine cylinder block automatic line rotary device based on the actual working conditions, and put forward an optimization scheme according to the analysis results, and get the optimal design point by using the DOE method to effectively reduce the mass, improve the intrinsic frequency, meet the strength and stiffness requirements of the device,

and achieve the purpose of reducing the cost, which provides a theoretical basis and method for the optimal design of this type of The optimal design of mechanism and equipment provides theoretical basis and method.
The DX module in the ANSYS Workbench environment can transfer the design variables of 3D models in actual engineering to the AWE environment. And using a finite number of design points to fit the response surface model, choose the suitable optimization algorithm to achieve the optimization effect of the structural design objectives, this function can be applied to the optimization processing of all kinds of 3D models in engineering practice.

AUTHORS' CONTRIBUTIONS

This work was carried out in collaboration among all authors. All authors read and approved the final manuscript.
Disclaimer (Artificial intelligence)

Option 1:

Author(s) hereby declare that NO generative AI technologies such as Large Language Models (ChatGPT, COPILOT, etc) and text-to-image generators have been used during writing or editing of manuscripts.

Option 2:

Author(s) hereby declare that generative AI technologies such as Large Language Models, etc have been used during writing or editing of manuscripts. This explanation will include list the name, version, model, and source of the generative AI technology and as well as the all input prompts provided to a generative AI technology

Details of the AI usage are given below:

1.

2.

3.

REFERENCES

1. Zhang Li. Factors affecting the performance of agricultural diesel engines and ways to improve the technology[J]. Agricultural Machinery Use and Maintenance,2022,(07):50-52.
2. LI Fan, ZHANG Shuli. Status quo and development trend of engine block cylinder head machining automatic line[J]. Automation technology of automobile production, 2014(5):35-37.
3. Yan Keying.Research on key technology of sand core flexible production line modification for cylinder block casting[D].Harbin Institute of Technology,2021.
4. BI Yan, FANG Changxing. Research on automatic loading and unloading device for engine cylinder block flexible production line[J].Manufacturing Technology and Machine Tools,2018(02):30-32.
5. MAO Jun, ZHANG Yunsheng, XIE Chunxue. Mechanical modeling of torsional vibration of scraper conveyor and analysis of over-curved section[J]. Mechanical Strength,2022,44(02):279-286.
6. Tang Yuanyu He Xiaoting. Application of finite element analysis in part structure optimization[J] Textile Machinery 2023,50(01):30-33.
7. Fang Can.Research on the application and trend of optimization design of mechanical structure[J] Science and Technology Innovation and Application,2016,166(18):137
8. Yang JX. Design and dynamics analysis of timing toothed chain drive system for gasoline engine[D]. Jilin University,2019.
9. ZHANG Liang,WANGXinyang,LIJian,WENZhengbiao,KONGLingyou. Design calculation of vertical roller spindle of oilsed soybean harvester[J]. Agricultural Development and Equipment,2020(02):83-84.
10. Guo Towelan.Design and research of automatic cutting machine for natural fiber[D].Supervisor:YanpingPeng;YongqiangJin.Dalian University of Technology,2019.
11. LI Min,TIANKaikun. Precise control of gear reduction synchronous motor based on ARM controller[J]. Journal of Hubei Normal University (Natural Science Edition),2022,42(03):42-47.

12. Meng Kun, Liu Xiaoli, Yu Zhifu et al. Optimization design of inner portal frame based on ANSYS Workbench[J]. Journal of Hebei Engineering University(Natural Science Edition),2014,31(01):98-101.
13. WANG Hongzhen,MENGQingrui,ZHENGDexi. Structural analysis and optimization of lifting device of agricultural milling machine based on finite element[J]. Research on Agricultural Mechanization,2019,41(09):40-44.
14. Tian Yuan, Zhong Fei, Yang Xiaojun et al. Design and finite element analysis of loading and unloading mechanism of cigarette filter rod[J]. Packaging and Food Machinery,2023,41(01):80-85.
15. YE Zhibiao.Research on overall structural design scheme of 2m pavement milling machine[J].Science and technology innovation and application,2014,(35):46-48.
16. Li Zhan-Yong. Parameter optimization of vibratory loading multilayer plate spring[D]. Yanshan University,2013.
17. LIU Jun,ZHANG Hong,HUANG Hua et al. Optimized design of mud pump crankshaft structure based on response surface model[J]. Journal of System Simulation,2019,31(05):879-885.
18. YAO Daozhuang,YANGJianguo,Lv Zhijun. Optimized design of shelf column section based on ANSYS Workbench[J]. Journal of DonghuaUniversity(Natural Science Edition),2011,37(04):438-441+475.
19. Mou Xiaohua. Research on structural design, optimization and control of finned tube palletizing robot [D]. Shanghai Ocean University,2016.
20. FENG Chao,JIANG Kai-Xin,WANG Ya-Hui et al. Multi-objective optimization of high-speed shaft of gearbox based on ANSYS analysis[J]. Machine Tools and Hydraulics,2020,48(20):139-143.
21. Cao Chongzhen, Jiang Peng, Wang Fengqin et al. Simulation analysis and optimization design of crank-rocker type cargo climbing vehicle[J]. Machine Tools and Hydraulics,2017,45(09):149-152.